DESIGN OF A HIGH HEAT FLUX FACILITY FOR THERMAL STABILITY TESTING OF ADVANCED HYDROCARBON FUELS

R. W. Bates, E. D. Maas, S. A. Irvine, T. P. Auyeung
Air Force Research Laboratory Propulsion Directorate
Edwards AFB, CA 93524

O. Peroomian and P. Kachgal
Metacomp Technologies, Inc.
Agoura Hills, CA 91391

ABSTRACT

With both the Air Force and NASA interested in developing reusable hydrocarbon-fueled engine technologies for reusable launch vehicles, there is an increased need to better understand the thermal stability of hydrocarbon fuels under high heat fluxes encountered during regenerative cooling. Currently, no existing thermal stability test rig can accurately simulate the high heat flux conditions that will exist in the cooling channels of these new high-pressure hydrocarbon engines. To meet the high reliability and reusability requirements proposed for these engines, the Air Force Research Laboratory (AFRL) has designed a High Heat Flux Facility (HHFF) using experience gained from past and present thermal stability test rigs. CFD++, a Metacomp Technologies Inc. computational fluid dynamics software suite, was employed to optimize the design prior to manufacture. Conjugate heat transfer calculations were performed in a single computational domain containing the copper heater block and the fluid channel of the new test rig design. The first tests conducted in the facility will be highly instrumented and will be used to validate the CFD++ calculations. The parameters of interest for a given geometry are the heat transfer coefficient, the degree of coking and corrosion in the channel, and the pressure drop as functions of heat flux, wall temperature, Reynolds number, channel material, fuel composition and pressure. The HHFF will be an important tool to facilitate the development and transition of new advanced hydrocarbon fuels under study by AFRL.

INTRODUCTION

Hydrocarbon fueled launch systems offer the potential for enhanced operability in a smaller vehicle, as compared to hydrogen fueled systems. Historically, smaller vehicles, both aircraft and rocket, have cheaper operating and maintenance costs. However, to realize these potentials, a hydrocarbon fueled launch system must be reusable and reliable. Performance can also be improved by increasing engine chamber pressures and by using higher energy density fuels. These increases also result in an increase of the heat flux across the engine walls, producing an even more challenging regenerative cooling environment. The fuels must now be able to withstand this increase in heat flux without producing undesirable deposits on the engine's cooling passages. To accomplish any of these goals, and especially to simultaneously accomplish performance increase and a high degree of reusability in hydrocarbon-fueled engines, a better understanding of the physical and chemical processes in the cooling channels is required.

The most simulative approach to study these processes would be to build a full-scale engine and produce actual conditions encountered during a launch. This is of course cost prohibitive, and fails to incorporate the timely and flexible modifications important during the initial stages of research and development. The key then is to find a way to closely simulate the actual conditions encountered by the fuel while maintaining a cost-effective and quantitative experimental apparatus. The conditions found in a regenerative cooling passage of a modern rocket engine include introduction of the fuel at high initial pressures (3000-7000 psi) and low initial temperature (from the initial vehicle tank temperature to slightly above)[1]. Fuel velocities vary as the cross-sectional area of the cooling channel does, providing a balance between channel pressure drop and the need for increased heat transfer to the fluid at the highest heat flux locations of the engine. The fluid mechanics caused by these variations can introduce secondary flow structures, which adjust convective heat transfer rates. Wall temperatures can vary from mild to pushing the factor of safety of the structural integrity of the wall material (500-1100 °F for copper walls). Heat flux

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE **14 APR 2004**	2. REPORT TYPE	3. DATES COVERED **-**

4. TITLE AND SUBTITLE **Design of a High Heat Flux Facility for Thermal Stability Testing of Advanced Hydrocarbon Fuels**	5a. CONTRACT NUMBER **F04611-99-C-0025**
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) **R Bates; E Maas; S Irvine; T Auyeung; O Peroomian**	5d. PROJECT NUMBER **4847**
	5e. TASK NUMBER **0244**
	5f. WORK UNIT NUMBER **48470244**
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **ERC Incorporated,555 Sparkman Drive,Huntsville,AL,35816-0000**	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S)
	11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution unlimited

13. SUPPLEMENTARY NOTES

14. ABSTRACT
With both the Air Force and NASA interested in developing reusable hydrocarbon-fueled engine technologies for reusable launch vehicles, there is an increased need to better understand the thermal stability of hydrocarbon fuels under high heat fluxes encountered during regenerative cooling. Currently, no existing thermal stability test rig can accurately simulate the high heat flux conditions that will exist in the cooling channels of these new high-pressure hydrocarbon engines. To meet the high reliability and reusability requirements proposed for these engines, the Air Force Research Laboratory (AFRL) has designed a High Heat Flux Facility (HHFF) using experience gained from past and present thermal stability test rigs. CFD++, a Metacomp Technologies Inc. computational fluid dynamics software suite, was employed to optimize the design prior to manufacture. Conjugate heat transfer calculations were performed in a single computational domain containing the copper heater block and the fluid channel of the new test rig design. The first tests conducted in the facility will be highly instrumented and will be used to validate the CFD++ calculations. The parameters of interest for a given geometry are the heat transfer coefficient, the degree of coking and corrosion in the channel, and the pressure drop as functions of heat flux, wall temperature, Reynolds number, channel material, fuel composition and pressure. The HHFF will be an important tool to facilitate the development and transition of new advanced hydrocarbon fuels under study by AFRL.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**		**12**	

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std Z39-18

rates varying from 1-100 BTU/in^2/sec, producing a range of boundary layer thickness and potential striations. Channel wall material variations, land thickness variations, surface roughness, and the asymmetric thermal loading of channel wall are all factors. Fuel composition, thermal properties, deposit formation and transport, potential material incompatibility and chemical corrosion of the channel surfaces, and fuel thermal history as it flows through the entire channel should be considered.

Resistively heated tube facilities (such as NASA Glenn Research Center's Heated Tube Facility [2-4], AFRL's Phoenix Rig, and the heated tube rigs of UTRC[5,6]and Rocketdyne[7,8]) and conduction heated thermal concentrators (such as Aerojet's Carbothermal rig[9,10]) have all been used to study the heat transfer behavior and thermal stability of fuels. Other options that can be used to generate the high heating rates found in rocket engine cooling channels are combustion, arc lamp and laser heating. All of these have their pros and cons for varying levels of simulation fidelity, complexity, control, and quantitative determination of results. The bulk of the data generated has been from heated tubes and the conduction heated thermal concentrator[1-12]. Direct ohmic heating of both pure metal and bimetallic co-annular tubes produces high wall temperatures and circumferential heat transfer to the fuel coolant. External wall temperature measurements and electrical power consumption are used to infer the internal wall temperature[2-8,11,12]. The implementation is relatively simple and allows for easy interrogation of post-tested specimens for coking and corrosion behavior. Power input requirements can be extreme, especially when the tube metal is copper or one of its alloys; and for this reason, typically heat fluxes achieved are less than 10-15 BTU/in^2/sec. Electrical isolation of thermocouples without loss of thermal contact or introduction of significant thermal resistance is critical. In the mid-eighties, Rocketdyne generated a heat flux of 84 BTU/in^2/sec using a bimetallic tube configuration[8]. This came at the cost of 4000 Amps current input and introduced extremely ambiguous results. Electrically driven chemical phenomena, as well as potential degradation of the tube materials, could not be ruled out as contributing factors in the results observed. Rocketdyne has also investigated asymmetric heating of a square tube using an applied heating strip on one side. Conditions studied ranged from 0.27 to 2.36 BTU/in^2/sec with wall temperatures approaching 500 °F. UTRC's heated tube work included bimetallic tube configurations and formed much of the early understanding of RP-1 coking and the role of sulfur compounds in copper corrosion[7,8]. NASA Glenn Research Center's Heated Tube Facility is actively investigating thermal decomposition and heat transfer behavior under rocket-like conditions up to 15 BTU/in^2/sec[2-4]. Channel curvature and aspect ratio effects have been studied as well[3]. The facility was recently modified in collaboration with AFRL to study sub-cooled hydrocarbon propellants.

Aerojet's Carbothermal Rig was based on a large copper conduction block internally heated by cartridge heaters[9,10]. The block was used to geometrically focus the thermal energy onto a small test section slab with a milled channel and braised closeout panel, which was simply machined off to expose the wetted wall for interrogation. Pressures of up to 3500 psi were studied using a variety of hydrocarbon fuels, including methane, propane, RP-1, and simulants of RP-1 doped with impurities. The rig generated heat fluxes of up to 53 BTU/in^2/sec and a range of heat transfer, coking, sulfur corrosion, channel coating and chemical refurbishment data. Unfortunately, it is no longer in existence.

Of the heat transfer rigs that have been used in the past, conduction heated rigs offer the closest simulation of the high heat flux conditions encountered during regenerative cooling of the rocket engine, with minimum complexity and maximum flexibility[9,10]. Asymmetric heating and easy instrumentation access of the channel are all advantages of this facility. Channel geometry and material can be freely adjusted without impact on the electrical input characteristics. Issues with the extremely large currents (> 4000 Amps) required by the resistively heated tube facilities to develop high heat flux and wall temperatures are avoidable. Thus, from the degree of simulation achievable, and the advantages of quantitative measurement and control of experiments, the conductively heated design was chosen. The Air Force Research Laboratory (AFRL) High Heat Flux Facility (HHFF) was designed as an extension of the Aerojet Carbothermal rig to achieve heat fluxes up to 100 BTU/in^2/sec.

COMPUTATIONAL DESIGN EFFORT

CFD++ is a computational fluid dynamics software suite developed by Metacomp Technologies Inc. It has been employed to perform fluid flow and heat conduction calculations on the new HHFF test

rig design. CFD++ has the ability to handle the conjugate heat transfer (CHT) problem between the solid (copper) test rig and the fluid flow within the test section. In this mode, the full Navier-Stokes equations are solved in the fluid, and a heat conduction equation is solved within the solid. The advantage is that complex geometries can be examined within a single computational domain by a single package.

Since the Aerojet rig was the basis for the HHFF design, the first phase was to reproduce, qualitatively, the rig's experimental results. Using CFD++, Aerojet experiments (propane fuel in small copper channels under high heat flux) were modeled, giving numerically predicted heat transfer coefficients that were within 10% of measured values. This problem was modeled using the full CHT mode in CFD++. With increased confidence in the CFD++ prediction capabilities for this class of problems, the Aerojet rig was modified to achieve higher heat transfer coefficients, as part of the next phase. A critical change was made to the Aerojet design based on these computations. The inlet and exit plenums showed significant potential for separation bubbles and recirculation of fluid near hot walls. Figure 1 shows velocity contours from that test channel. Realizing that an extension of the Aerojet design must accommodate high velocity channel flows to sustain higher heat fluxes, the test section was redesigned to incorporate straight-through flow with inlet and outlet diffusers that avoid separation for the highest test section velocities. Figure 2 shows the result of that redesign.

In a two step process, CFD++ was used to quickly test the proposed rig design modifications. In the first phase, a two-part analysis was used. In one part, fluid flow through the pipes and test section was simulated to make certain that there was no (or minimal) separation in the flow. In part two, the heat conduction through the solid was solved independently, with the fluid effect modeled as a convective heat transfer rate boundary condition. This allowed for iteration of the design concept with minimal cost and rapid feedback. Results from this study revealed the design that would obtain the most uniform temperature in the inner core of the test section. Using these simple results, the next step was to validate the uncoupled results by performing a complete CHT simulation of the final design. Using CFD++ this way allowed the effects of any design modifications to be determined before beginning the construction of the test rig.

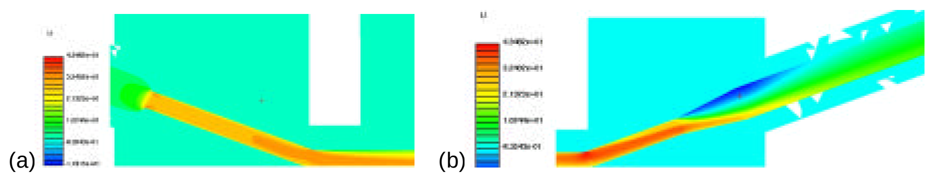

Figure 1: Velocity (m/s) profiles showing separation in a representation of an Aerojet (a) inlet and (b) exit

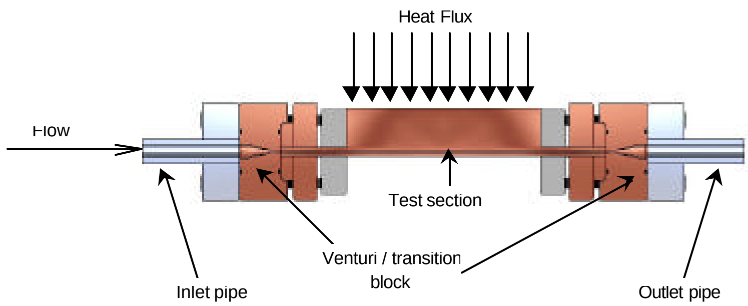

Figure 2: Test section diagram showing redesigned flow channel

FLUID FLOW PROBLEM

An accurate representation of the fluid flow in and out of the test rig was accomplished by modeling the fluid/solid interaction via a simple heat transfer boundary condition. The 3-D geometric model composed of an inlet pipe, a converging venturi, a square test section with cross sectional area 4.0 x 10^{-4} in^2, a diverging venturi and an outlet pipe. Earlier results were used to make the following improvements to the channel design:

1. In order to decrease the velocity ratio between the flow into the inlet and through the coupon portion, the inner diameter of the inlet/outlet pipe was reduced to 0.12 in.

2. Length of the diverging venturi portion of the pipe was increased to eliminate the flow separation.

A hexahedral mesh consisting of 176,128 cells was used to conduct a steady state calculation. The simulation utilized properties of liquid propane with the following boundary conditions:

1. inlet mass flow rate = 0.02 kg/s and temperature at 160 K

2. back pressure in the outlet

3. convective heat transfer wall conditions of T_8 = 1400 K and a heat transfer coefficient of 400 W/m^2K were set on the test section upper surface

The steady state results showed no flow separations, and further validated the design improvements.

SOLID HEAT CONDUCTION PROBLEM

The heat conduction, for three different configurations of the test rig, was solved for using CFD++. The configurations were identical except for the shape of the thermal mass mounted on the coupon and the number of heaters as shown in Figure 3.

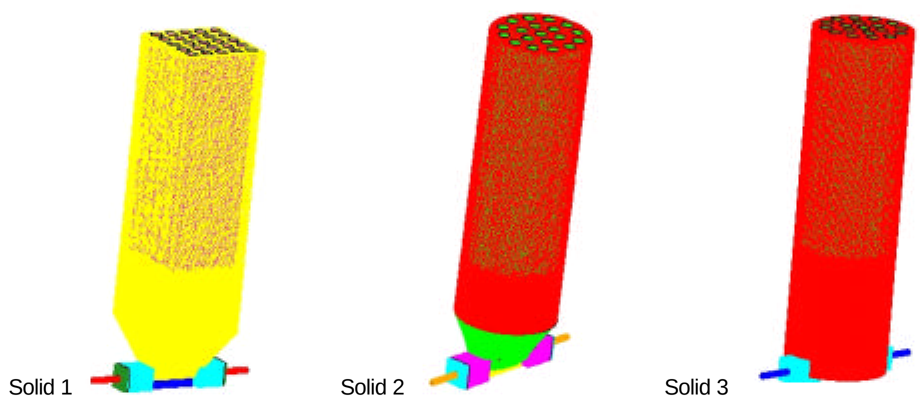

Figure 3: Solid model variations of the heater block

Solids 1 and 2 were geometrically focused at the bottom. This design feature was meant to allow rapid replenishment of the heater's thermal reservoir region nearest the test section. Solid 1 had a square cross section, while Solids 2 and 3 had circular cross sections. Only Solid 3 incorporated an embedded test section, similar to the original Aerojet Carbothermal rig. The heater length for all three versions was 10 in. Each design had a sufficient thermal diffusion region below the cartridge heater section to allow for smooth thermal wave propagation to the test section. A tetrahedral mesh for each of the configurations

was made up of 4,381,926, 3,092,415, and 4,864,786 cells, respectively. The heat conduction equation was solved for each configuration using CFD++ and the following initial conditions.

$K = 395\ W/m^2K$ (pure Cu)
$T_{initial} = 300\ K$

Surfaces of the inner inlet/outlet pipes, converging and diverging venturi, and inner test section used a convective heat transfer coefficient of $400\ W/m^2K$ and $T_8 = 160\ K$ as the boundary wall conditions. Heater surfaces were set as isothermal walls with T=1400 K. All other surfaces were set as adiabatic walls. Based on the temperature distribution results at the inner core of the test section (Figure 6), it was apparent that Solid 3 would provide the greatest heat transfer rate to the fluid; however, Solid 1 provides for a more uniform upper wall temperature.

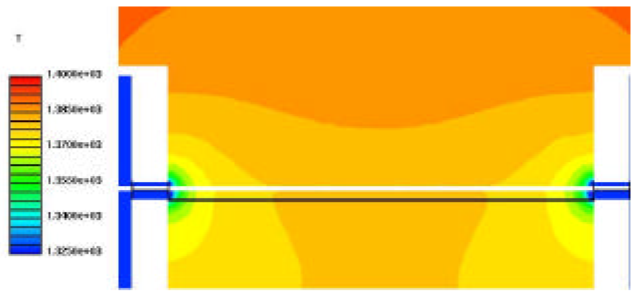

Figure 4: Solid 1 - Steady state temperature (K) profile of coupon section on XZ cutplane (y=0m)

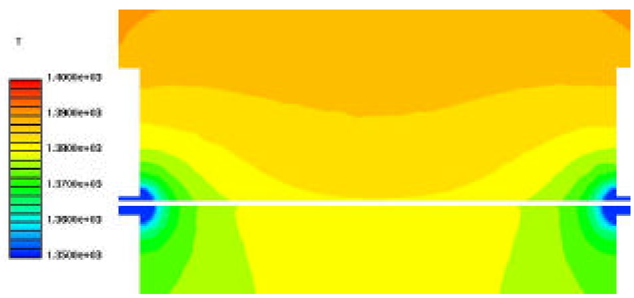

Figure 5: Solid 2 - Steady state temperature (K) profile at coupon section on XZ cutplane (y=0m)

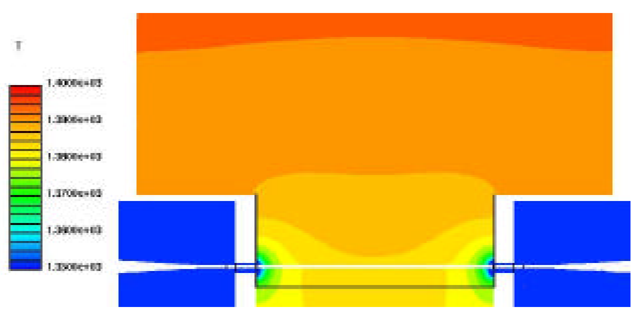

Figure 6: Solid 3 - Steady state temperature (K) profile of coupon section on XZ cutplane (y=0m)

FULL CONJUGATE HEAT TRANSFER PROBLEM

Based on the results of the simple solid heat conduction models, as well as further modeling with higher heat transfer coefficients, a hybrid geometry of Solids 1 and 3 (as shown in Figure 7) was selected for the final design. This design incorporated the strengths of geometric focusing, which improves the uniformity of upper wall temperature by effecting faster thermal replenishment, with embedding of the test section in the heater block, which increases the thermal mass in proximity to the test section. Both heat flux and temperature uniformity are improved as a result. Also, the length of the solid copper section of the heater block (section directly below the cartridge heater locations) was reduced. It was determined from previous designs that the section was longer than needed to achieve thermal smoothing from the cartridge heater section. A full conjugate heat transfer analysis was performed. Temperature contours are shown in Figure 8, (a) and (b), and emphasize the benefit of higher thermal mass near the test section with focusing diffuser sections.

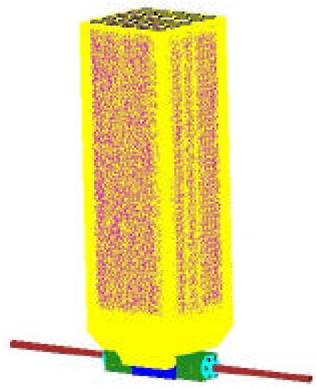

Figure 7: Final Geometry showing complete grid with both hexahedral and tetra meshes

(a). (b).

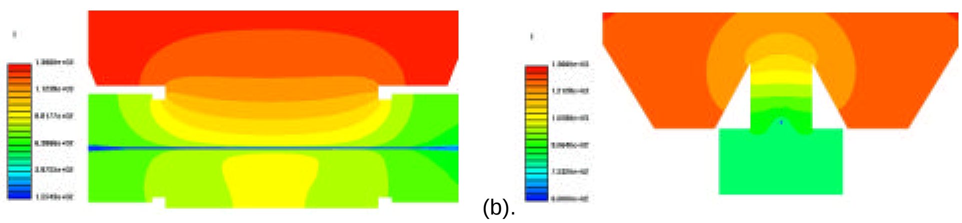

Figure 8: (a) Temperature (K) contour plot on YZ plane at x=0 (b) Temperature (K) contour plot on XY plane (z=0)

FACILITY DETAILS

With confidence in the ability of the test section and heater block design to achieve uniform, high heat flux, the design of the supporting facility was undertaken. The overall goal of the HHFF is to accurately simulate the conditions fuels encounter while cooling rocket engine passages as realistically as possible. Maximum test section pressures of 4500 psi and maximum wall temperature of 1200 °F are also desired. These maximums were set based on the compromise between component cost and availability and the desire to keep simulative flexibility for a range of fuels. These conditions are significantly higher than current facilities are addressing.

The test section is designed to allow for flexibility of channel geometry. Test section lengths can vary from two to six inches with channel widths from 0.020 to 0.090 inches and channel heights from 0.020 to 0.200 inches. There is also potential for multi-channel test sections with varying land widths. Figure 9 shows 3-D views of the test section assembly along with a 2-D cross-sectional view. The assembly includes a transition section from the circular tubing to the square test channel.

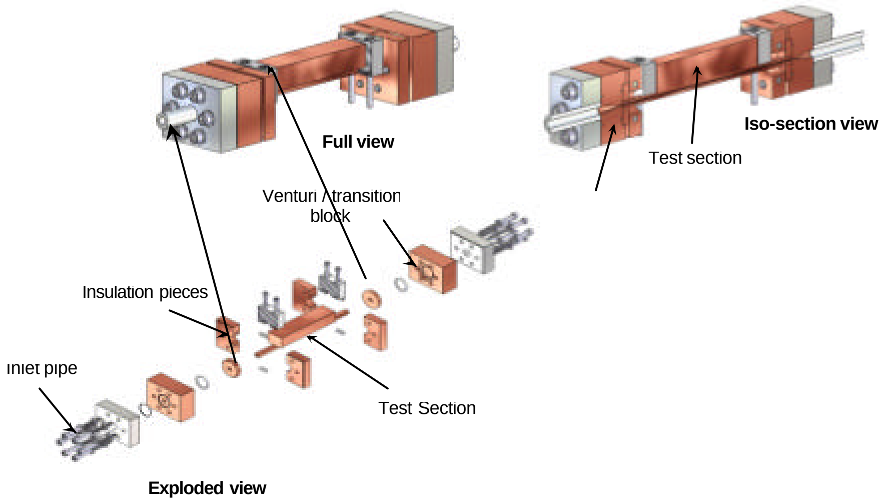

Figure 9: Test Section Assembly Design (Full view, iso-section view, and exploded view)

The test section, along with the copper heating block, are enclosed in an altitude chamber in order to reduce convective losses, and subsequently, allow for more quantitative heat transfer calculations. The chamber assembly is shown in Figure 10.

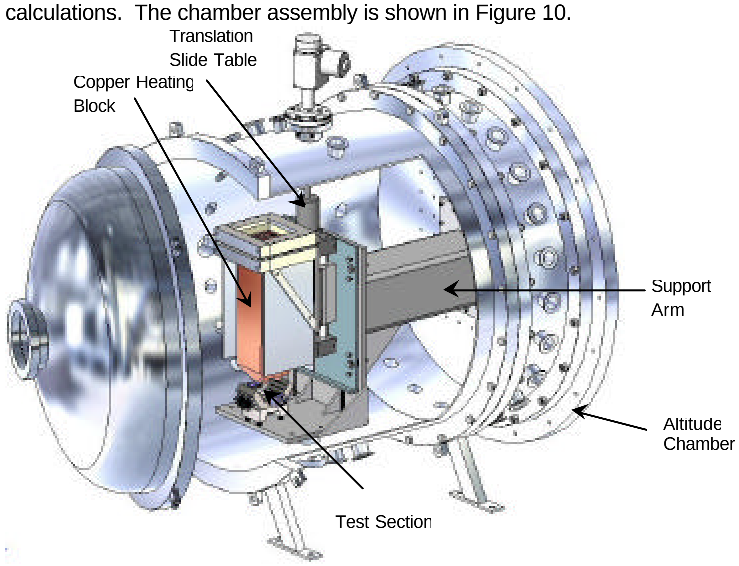

Figure 10: Altitude Chamber Configuration

The heater block, containing nominally 25 cartridge heaters, will be heated up and then lowered directly onto the test section using a linear slide table. This will allow both the fluid flow within the fuel channel and the copper heater block to reach steady state before any contact occurs. Having the copper block mobile also allows for the contact between the block and the test section to separate once the test is complete, eliminating residual heat transfer from the block to the test section during shutdown. In this manner, transient behavior during startup and shutdown of a given test can be controlled.

In order to accurately measure the temperatures of the test section and heater block a highly instrumented test section cradle was designed (Figure 11). The material chosen for the cradle is aluminum in order to ensure rigidity and ease of manufacture of the pieces. The cradle sides mimic the shape of the thermal reservoir to allow proper measurement of the thermal wave passing down to the test section. These temperatures are measured by spring-loaded thermocouples located throughout various slots drilled through the aluminum support cradle sides and along the bottom. The location of the thermocouples on the bottom was determined due to the small size of the test section coupled with the high velocities of the fuel. It was feared that placement of thermocouples within the channel would disturb the flow and cause unpredictable fluid behavior during the tests as well as possibly affecting the corrosion and deposition of the system.

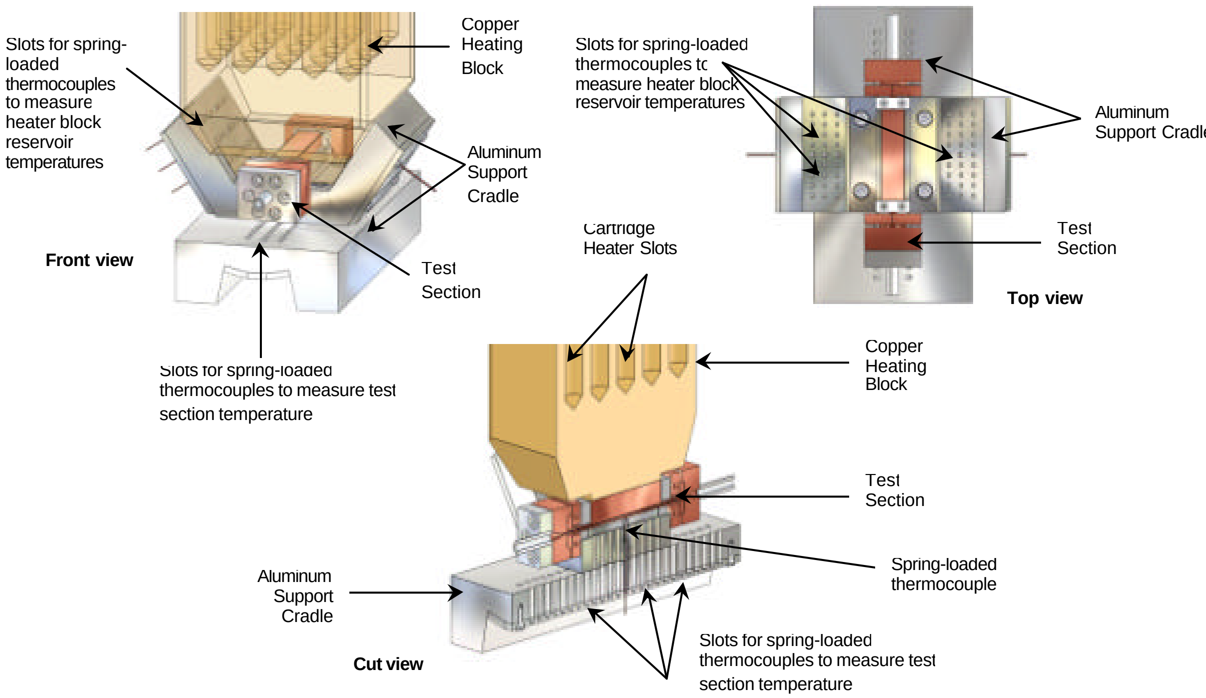

Figure 11: Front, Top, and Cut Views of Test Section Cradle

To ensure the test section pressures reach the desired 4500 psi, it is necessary to have system inlet pressures around 6000 psi. Bladder tanks are being used for fuel storage and pressurization in order to eliminate diffusion of nitrogen into the fuel. A Coriolis flow meter, accurate to 0.10% of total flow, is used to measure the mass flow rate and density of the fuels.

A preheater is utilized to control the inlet bulk temperature to the test section, which allows greater flexibility in producing varying thermal history of the fuel. The preheater consists of a specially

designed dual tube, coiled heat exchanger cast-in with six groups of 4 KW heating elements inside an aluminum block. Using either the coiled tube heat exchanger or the cast-in heater elements, the fuel temperature can be controlled from –4 °F to 500 °F. The heater elements are configured into three zones for multi-level heating control to prevent hot spots from causing fuel coking and deposition inside the cast-in heat exchanger. In this mode of operation temperatures up to 500 °F can be produced. The preheater also has the ability to function in parallel or counter-flow with its coolant tube coiled around the fuel tube. In this mode, a glycol recirculating system will be used to provide continuous heating or cooling in the low to mid-temperature range from –4 °F to 176 °F. This system is critical in maintaining constant fuel temperature near room temperature without being affected by the temperature changes of the surroundings.

Mass flow through the test section is controlled by a cavitating venturi located between the preheater and the test section. This eliminates the potential for feed system coupled oscillations that could affect the flow through the test section.

Located following the test section and outside the altitude chamber are several data collection and control apparatuses. One of these is a collection filter for particles discharged from the test section. Also, a sample vessel will be attached for fluid collection during experimentation. Samples collected from both of these will be used for additional studies of fuel degradation under extreme conditions. Following the sample vessels, a compact, lightweight electronic control valve designed for precise fractional flow control will be employed for regulating the system backpressure. Pressure transducers and thermocouples are placed throughout the system for both data collection and observation of behavior in the channel.

To provide control over dissolved oxygen content, a sparging system will be used. A sparging tank, consisting of a GN_2 feed, a giant stir rod, and several oxygen probes will be used to reduce dissolved oxygen in the fuel. After fuel is placed in the sparging tank, the GN_2 feed will be activated, and a giant stir rod will agitate the fuel to avoid potential striation. This is possible since the tank is sitting on a large-volume stirrer, rotating the stir bar between 60 and 1050 rpm with the capacity to stir a maximum volume of 300 lbs. Oxygen probes will be located at various intervals along the sides of the tank and are capable of detecting dissolved oxygen as low as six parts per billion. This allows for study of the effects of dissolved oxygen and fuel additives designed to reduce thermal-oxidative decomposition.

In the High Heat Flux Facility, a programmable logic controller (PLC) is used to provide the overall operational integrity of the facility in five major areas. These include remote valve operation, maintaining fuel flow, fuel heating and cooling processes, and initialization of data recording. This is achieved by utilizing a combination of digital and analog interface modules with advanced logic instructions for intelligent control of the facility. In parallel with the PLC, a data acquisition system and two heater control systems operate simultaneously. The data acquisition system, PI 6000, manufactured by Pacific Instruments, is a mainframe system with integrated signal conditioning providing 16-bit measurement resolution. Currently, a total of 80 analog input channels are set up for facility pressure and temperature measurements. A digital interface module with 16 input and 16 output channels is also added to the system for communication with the PLC and other compatible digital interface devices. A 32-zone temperature controller panel, custom-built by Watlow, is responsible for zone-independent high heat flux condition generation within the test section. In addition, a built-in wattmeter is set up to measure the total power consumption. Global alarm outputs from this controller are directly connected to the PLC for safe operation.

A variety of software applications were evaluated for integration with the facility hardware. Required specifications, operational needs, and stability were all key factors in selecting the final configuration. RSView32, a human-machine interface (HMI) type application developed by Rockwell Automation, was selected for facility control. PI 660, the data acquisition software developed by Pacific Instruments for use with their mainframe data system, was the main reason for selecting the Pacific Instruments PI 6000 data acquisition system. Software integration between the two applications is accomplished by sharing real-time data collected from the PI 6000 with RSView32 for display and PLC process control. Moreover, an add-on health monitoring program is integrated into the PI 660 software to

monitor critical parameters for out of limit conditions. Both data and control systems are designed with expandability in mind, giving the facility flexibility for future modifications. Lastly, Anawin is a software package developed by Watlow to accompany the temperature controller. It supports controller functions such as, monitoring thermal distribution and maintaining high heat flux conditions using either built-in PID control functions or by predetermined ramp and soak profiles. An overview of the entire data acquisition and control system architecture is shown in Figure 12.

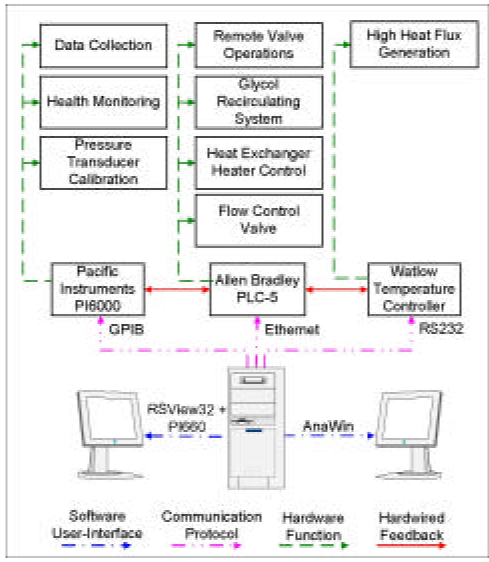

Figure 12: Data Acquisition and Experiment Control System

TEST AND ANALYSIS

Using the RSView32 Graphical User Interface (GUI) created for the facility, test operations are controlled remotely. A typical test begins with the test section installed in the altitude chamber under a vacuum. The heater block starts a few inches above the test section, out of thermal contact. Cartridge heater temperatures are pre-set and the entire heater section is allowed to reach steady state. Fuel flow is started at this point, and the pre-heater is activated. When a steady state for the fuel inlet condition is reached, the heater block is lowered onto the test section via the linear slide table. Once it makes thermal contact with the test section, time zero for the test begins. In this way, initial transients are minimized and the fuel never sees a wall temperature higher than during its planned test time. This is critical to quantitative analysis of the post-test results. Test durations ranging from ten minutes to one hour are nominal, depending on the specific test requirements. Throughout the test, facility-set parameters, such as cartridge heater temperature, electrical power input, mass flow rate, and inlet pressure and temperature, are monitored. Test section temperatures and pressure drop are observed as an early indicator of coking and/or flow blockage. When the test duration is met, the heater block is lifted off the test section to break thermal contact, allowing the test section to cool. Fuel flow continues to allow for faster cooling of the test section and to prevent thermal soak back and coking of stagnant fluid on the test section walls. Fuel flow is stopped and the test section is purged with nitrogen when the test section temperature falls to an acceptable level. Once the test section and heater block have cooled, the test section will be removed for further analysis.

For internal wetted surface analysis, each test section will be cut apart and sectioned to expose the fluid channel. Each individual section will then be examined by three different methods. Some

sections will be examined by scanning electron microscope (SEM) to determine morphology of deposition, chemical attack of the channel walls, and diffusion of species into the substrate. Then, during SEM analysis, Energy Dispersive X-Ray scattering (EDX) will be used to determine composition of deposits, examine grain boundaries, and examine exposed metal surfaces. Lastly, carbon deposition rates will be determined by LECO analysis of approximately ¼-inch sections of the test channel. In addition to enabling understanding of coking behavior of the fuels, these analyses will also demonstrate interaction of various compounds, including sulfur-containing compounds, with the wetted materials of the test channel. This will help quantify corrosion of fluid channels by sulfur type. The results will also aid in determining and comparing thermal stability of both current and newly created fuels.

A post-test analysis of all measurements obtained during experimentation, including pressures, temperatures, mass flow rates, and electrical energy supplied, enables quantitative calculation of experiment conditions. Determinations of overall heat transfer coefficient (h_c), pressure drop (ΔP) along the test channel, axial temperature profiles throughout the length of the section will help establish wall and core bulk temperatures (T_w and T_b respectively) for each test. These parameters are of value in determining how efficient a fuel will be for use in cooling. Comparisons will be made with computational modeling of each experiment using CFD++. Particular attention will be paid to the very thin boundary layers encountered during high heat flux cooling.

Along with varying the flow rate and type of fuel used, the test section material will change as well. Some of the materials being tested in this system are Haines 188 and 214, GRCop84, NarloyZ, and OFHC copper. By testing these materials the chemical interaction between the fuels and the channel material will be identified, along with the test section material's tolerance of the high temperatures and pressures of the facility.

SUMMARY AND CONCLUSIONS

The design of a unique thermal stability facility was undertaken to facilitate the development and transition of advanced hydrocarbon fuels. The High Heat Flux Facility (HHFF) is capable of achieving 100 BTU/in^2/sec and operates up to 4500 psi, closely simulating the actual conditions found in the cooling passages of a modern rocket engine. Using experience gained from past and present thermal stability test rigs in conjunction with CFD++, a Metacomp Technologies Inc. computational fluid dynamics software suite, conjugate heat transfer calculations were performed in a single computational domain containing the copper heater block and the test section fluid passages to optimize the design. A description of the facility and test parameters of interest (the heat transfer coefficient, the degree of coking and corrosion in the channel, and the pressure drop as functions of heat flux, wall temperature, Reynolds number, channel material, fuel composition and pressure) was given. Asymmetric heating, easy instrumentation access, and flexibility of the channel geometry are all advantages of this facility.

FUTURE WORK

The HHFF will be used in the future to characterize the thermal performance, coking and corrosion behavior of RP-1, new grades of RP-1, new advanced synthetic hydrocarbon rocket propellants, sub-cooled hydrocarbons propellants, and additives for enhanced thermal stability, storability, and performance. The facility may also be used to study the effects of channel geometry, surface roughness, coatings, channel materials, and heat-transfer aids such as riblets, dimples, and bumps.

ACKNOWLEDGMENTS

The authors would like to thank Mr. Philip Rice, Ms. Claudia Heflin, and Mr. Wesley Mosier of Applied Engineering Services, Inc. (AES), Mr. R. Mike Griggs, Mr. Earl Thomas, Mr. Randy Harvey, and Mr. Toya Johnson of Engineering Research and Consulting, Inc. (ERC), Mr. Mark Wilson of ERC, Inc. (ERC/ROSS), Mr. Mark Pilgram and Mr. Matt Kracke of Sverdrup/ROSS for their support throughout the course of this project. The authors also wish to thank and acknowledge the technical discussions and advice of Major Abdollah Nejad (AFRL/PRS) and Mr. Jay Levine (AFRL – Aerophysics Branch Chief).

REFERENCES

1. Bates, R. W., Edwards, J. T., and Meyer, M. L., ***Heat Transfer and Deposition Behavior of Hydrocarbon Rocket Fuels***, Aerospace Sciences Conference AIAA 2003-0123, Reno, NV (Jan 2003)

2. Green, J. M., Pease, G. M., and Meyer, M. L., ***A Heated Tube Facility for Rocket Coolant Channel Research***, 31st Joint Propulsion Conference and Exhibit AIAA-95-2936, San Diego, CA (Jul 1995)

3. Meyer, M. L., ***Electrically Heated Tube Investigation of Cooling Channel Geometry Effects***, 31st Joint Propulsion Conference and Exhibit AIAA-95-2500, San Diego, CA (Jul 1995)

4. Stiegemeier, B., Meyer, M. L., Taghavi, R., ***A Thermal Stability and Heat Transfer Investigation of Five Hydrocarbon Fuels: JP-7, JP-8, JP-8+100, JP-10, and RP-1***, Joint Propulsion Conference and Exhibit AIAA 2002-3873, Indianapolis, IN (Jul 2002)

5. Giovanetti, A., Spadaccini, L., Szetela, E., ***Deposit Formation and Heat-Transfer Characteristics of Hydrocarbon Rocket Fuels***, Journal of Spacecraft and Rockets, Vol. 22, No. 5, 1985

6. Marteney, P., Colket, M., Vranos, A., NASA CR-168027, ***Experimental Study of the Thermal Stability of Hydrocarbon Fuels***, Contractor Report, East Hartford, CT (Feb 1983)

7. Hines, W. S., ***Heat Transfer to RP-1 Kerosene in Turbulent Flow Under Asymmetric Heating Conditions***, Chemical Engineering Progress Symposium Series, Heat Transfer, pp 193-200

8. Cook, R. T., and Kirby, F. M., NASA CP-2437**,** ***Advanced Earth-to-Orbit Propulsion Technology 1986*****,** Vol. 2, pp 461-473, 1986.

9. Gage, M. L., Rosenberg, S. D., NASA CR-185203, ***Hydrocarbon-Fuel/Combustion-Chamber-Liner Materials Compatibility***, Interim Final Report, Cleveland, OH (Apr 1990).

10. Rosenberg, S. D., Gage, M. L., Homer, G. D., and Franklin, J. E., ***Hydrocarbon-Fuel / Copper Combustion Chamber Liner Compatibility, Corrosion Prevention, and Refurbishment***, Journal of Propulsion and Power, Vol. 8, No. 6 (Nov-Dec 1992)

11. Striebich, R. C., Lawrence, J., ***Thermal Decomposition of High Energy Density Materials at High Pressure and Temperature***, Journal of Analytical & Applied Pyrolysis, 70, 339, 2003.

12. Wohlwend, K., Maurice, L. Q., Edwards, T., Striebich, R. C., Vangsness, M., Hill, A. S., ***Thermal Stability of Energetic Hydrocarbon Fuels for Use in Combined Cycle Engines***, Journal of Propulsion and Power, Vol. 17, No. 6, (Nov-Dec 2001)